Renewable and Sustainable Energy Reviews 22 (2013) 539–549

Contents lists available at SciVerse ScienceDirect

Renewable and Sustainable Energy Reviews

journal homepage: www.elsevier.com/locate/rser

Numerical modelling of ventilated facades: A review

Alvaro De Gracia*, Albert Castell, Lidia Navarro, Eduard Oró, Luisa F. Cabeza

GREA Innovació Concurrent, Edifici CREA, Universitat de Lleida, Pere de Cabrera s/n, 25001-Lleida, Spain

ARTICLE INFO

Article history:
Received 11 June 2012
Received in revised form
11 February 2013
Accepted 18 February 2013
Available online 15 March 2013

Keywords:
Review
Ventilated facades
Double skin facades (DSF)
Buildings Numerical modeling

ABSTRACT

The use of double skin facades (DSF) in the building sector and its thermal benefits have been widely studied numerically over the last 30 years. These modelling and simulations are based on different typologies, which have evolved altogether with the available computational resources. The models that have been used to study the thermal performance of DSF can be grouped as analytical and lumped models, non-dimensional analysis, network models, control volume, zonal approach, and computational fluid dynamics (CFD). This paper describes these different typologies of numerical modelling highlighting their benefits and limitations, and overviews the research produced using each typology.

Contents

1. Introduction

Nowadays, the building sector is consuming 40% of the global energy in the European Union, being two-thirds of this energy consumption due to the heating, ventilating and air conditioning (HVAC) systems. In order to reduce this high energy demand, the European directive on the energy performance of buildings (EPBD) suggests that all the EU member states should approve energetic policies to promote the inclusion of very low and even close to zero energy buildings [1].

The improvements in buildings envelopes have high potential in energy demand reduction and consequently in energy savings [2]. Within this context, the use of double skin facades (DSF) in the building sector has recently become more popular. Those facades, if well designed, can efficiently reduce the overall HVAC consumption in buildings by absorbing part of the solar radiation during winter and preventing overheating during warm periods [3]. Moreover, the use of ventilated DSF can improve the acoustic characteristics and day lighting inside the building.

Those constructive systems are based on a special type of envelope, where a second skin, usually a transparent glazing, is placed in front of a regular building facade. The air space in between (the channel) can be mechanically or naturally ventilated to improve the thermal performance of the building [4]. The DSF can work under different ventilation modes (Fig. 1a–c) or work as a Trombe wall (Fig. 1d), depending on the energetic requirements and weather conditions.

In addition, in the systems with external and internal glazing, an adjustable sunshade device is usually installed in the channel to prevent overheating during cooling periods [5]. Even though

* Corresponding author. Tel.: +34 973 3576; fax: +34 973 3575.
E-mail address: lcabeza@diei.udl.cat (A. De Gracia).

1364-0321/$ - see front matter
http://dx.doi.org/10.1016/j.rser.2013.02.029

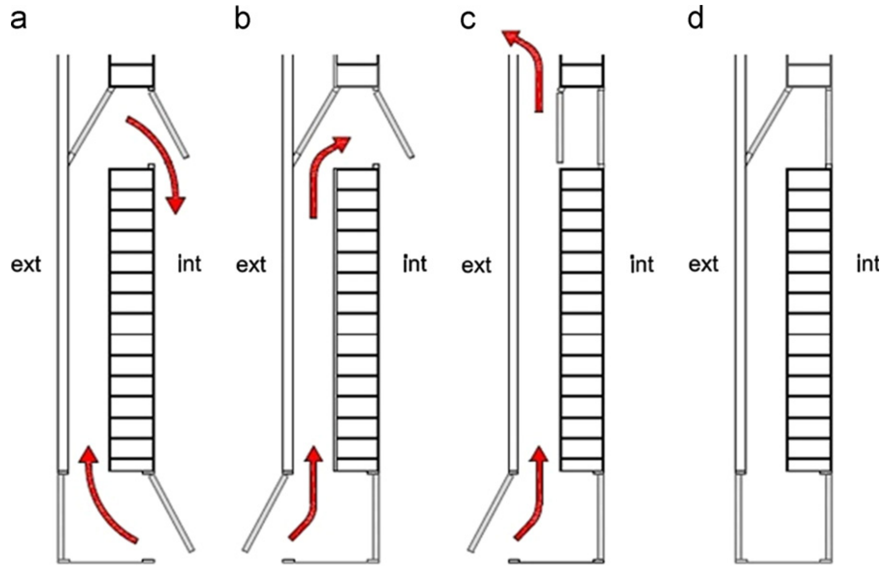

Fig. 1. Different operational modes of a ventilated DSF.

there are several different typologies of ventilated DSF, they are usually grouped under box window facade, shaft-box facade, corridor facade, and multi-story facade [6–8].

Ventilated facades and their thermal behaviour have been a topic of research over the last years. Shameri et al. [3] reviewed the literature related to DSF concluding that ventilation has been widely studied, while lack of research has been carried out regarding to the day lighting capacity of the envelope. The high fire hazard risk of this type of envelopes and the need of more research for better understanding of the DSF effects was also mentioned.

Moreover, Zhou and Chen [9] reviewed the potential of using DSF in the hot-summer and cold-winter zone in China, concluding that the implementation of this type of envelopes would be an efficient way to meet the task of sustainable commercial buildings design in China, if controlled shading devices are applied.

The available literature on the field is specially focused on studying numerically the thermal performance of the DSF under different climate zones and different ways of implementation. Nevertheless, other than the numerical studies, important effort has been done in the experimental field for better understanding the behaviour of these systems and to provide data and tools which might be used to validate the numerical models developed [10–14].

Numerical models are becoming essential in the design phase of these complex constructive systems. Models used to study numerically the thermal performance of the DSF systems can be grouped as analytical and lumped models, non-dimensional analysis, network models, control volume models, zonal approach, and computational fluid dynamics (CFD). The main goal of this paper is to overview the different typologies of numerical models and to describe the thermal response of these systems.

2. Analytical and lumped models

Analytical and lumped models can generally provide specific useful information in the design phase of the DSF without consuming high computational resources. However, several hypothesis must be assumed to solve the analytical models, while the lumped models assume constant temperature at each facade and cavity of the DSF.

The first numerical models which were developed to study numerically the behaviour of a DSF correspond to simple one-dimensional analytical models which solve the first principle using empirical correlations. In [15] the Trombe wall is treated as a solar flat plate collector taking into account its particularities, concluding that the most important factor in the performance of the Trombe wall is the transmissivity of the cover. Holmes [16] analysed and optimized in 1997 some of the design parameters of a naturally ventilated glazed facade using an analytical model which assumes linear vertical temperature gradient. The thermal transmittance in steady state and the flow rate due to natural convection in the air gap were studied in systems with one or two glazed layers with different emissivities.

It is well known that flow characteristics of DSF channel play a key role in the performance of the system. Grabe [17] developed and validated a simple simulation algorithm based on energy transport and Bernoulli equations to study the thermal behaviour and flow characteristics of DSF. The model proves the sensitivity and the difficulty of modelling flow resistances in the air channel.

Ciampi et al. [18] presented an analytical method based on the electrical analogy to calculate the electrical energy savings in buildings due to the use of ventilated facades in Southern Europe climates during summer. The model was used to analyse two particular cases. In the first case, inner masonry wall was given and the authors optimized the air duct and the outer facing. On the other hand, in the second case, the outer facing was fixed and, air duct and inner masonry were parameterized. It was concluded that in all cases the energy demand decreased with the air duct width and solar radiation. It was stated that the use of well-designed ventilated facades in buildings can reduce the electricity consumption for summer cooling by more than 40%.

A physical model also based on the thermal resistance network was proposed by Ong [19] to analyse the thermal performance of a solar chimney, similar to the Trombe wall. Steady-state heat transfer equations were solved using a matrix-inversion solution procedure. In this simple model the temperatures at all surfaces were assumed uniform, and air inlet temperature was supposed to be equal to the room temperature. Empirical correlations available from the literature were used to calculate convection and radiation heat transfer coefficients in the model, which proved good agreement with experimental data from Hirunlabh et al. [20].

Park et al. [21,22] optimized the energy performance, the visual comfort, and the thermal comfort of a DSF by using a motorized louver slat in the cavity and ventilation openings. Here, a two-dimensional lumped simulation model was developed under different operational modes. This numerical model, instead of using

empirical correlations available in the literature, was calibrated based on a parameter estimation technique using in situ measured experimental data. It was demonstrated that in lumped models based on descriptions of physical processes and augmented by calibration parameters to deal with the assumptions (surface temperatures, constant convective heat transfer coefficients, etc.), the calibration process plays an important role in the performance of the model and improves the accuracy of the models in comparison to experimental data.

3. Non-dimensional analysis

Balocco used non-dimensional analysis to determine the thermal performance of a naturally [23] and mechanically [24] ventilated facade. This methodology applies the Buckingham theorem to create correlations depending on non-dimensional numbers, so the same parameters might describe the process at different scales.

For the naturally ventilated DSF [23], 14 non-dimensional numbers with physical meaning were used to create a correlation based on experimental data, and to determine the heat flux transferred to the inner environment throughout the wall. On the other hand, 12 non-dimensional parameters were used to describe the thermal performance of the mechanically ventilated facade [24]. Both correlations proved to be valid for a wide range of conditions and were validated using experimental data and CFD simulation results. The non-dimensional models assume constant thermo-physical properties, except for air density, and they are presented as basic tools to evaluate some specific parameters which might be useful to design ventilated facades without using high complex simulation programs.

4. Airflow network modelling

Hensen et al. [25] stated that the airflow network method treat every building component and relevant HVAC fluid flows systems as a network of nodes representing rooms, parts of rooms, and system components, with internodal connections representing the distributed flow paths associated with cracks, doors, pipes, pumps, ducts, fans, and the like. Conservation of mass for the inlet and outlet flows of each node leads to a set of simultaneous and non-linear equations, which are integrated over time to characterize the flows. This airflow network modelling can provide fast useful information about bulk flows without consuming high computational resources. This method uses pressure differences and discharge coefficients for simple cross ventilation and network flow analysis where multiple inlets and outlets and internal flow branching occurs [26]. However if details about the nature of the flow field are required, CFD simulation must be used.

The airflow network model is usually integrated with a thermal network, which solves the heat balance in each node [9]. The coupling between the thermal and airflow model will be further discussed in the present paper. Tanimoto and Kimura [27] studied a DSF with venetian blinds using thermal and airflow networks (Fig. 2). Good agreement between the calculated pressures differences and temperatures and the experimental data was found.

Moreover, when the network modelling is used to analyse the thermal performance of a DSF integrated into a building, a thermal model of the building must be included in the algorithm altogether with the airflow model and the thermal model of the DSF [28]. The airflow model used in the DSF must take into account the buoyancy effect caused by thermal gradient along the channel and wind effect which may create pressure differences between the inlet and outlet. Stec and van Passen [29] used the network model

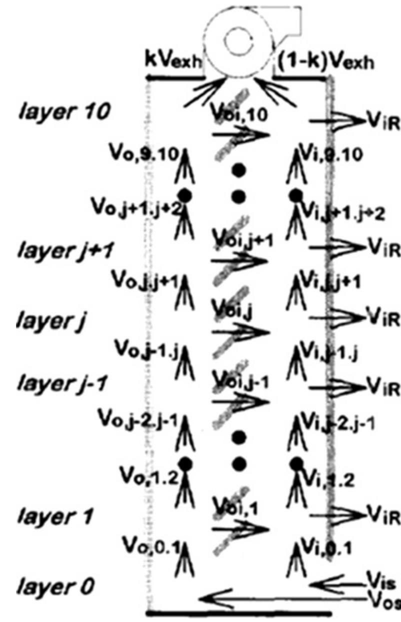

Fig. 2. Airflow network model applied to a ventilated facade with an integrated roll screen [27].

to settle down that to design a building with DSF, the integration of the HVAC system should be together with the facades. The authors developed a simulation program using Simulink to compare the thermal behaviour and the economic costs of different facades and HVAC solutions. DSF with external shading was chosen as the most promising technology for Dutch weather conditions. In addition, the same software was used to highlight the benefits of using plants instead of blinds in the DSF for shading and solar absorption purposes [30]. The study showed that temperature at all locations is reduced, as well as the cooling loads, when plants are used in the cavity. Moreover, the thermal performance of a DSF in hot and humid climates were modelled by Haase et al. [31] using thermal building simulation (TRNSYS) linked to a nodal airflow network (COMIS). The model was validated with experimental data, and was used to evaluate the impact of different factors in the cooling energy consumption of the building, such as orientation, window to wall ratio, and glazing types.

Fallahi et al. [32] described numerically an innovative design of DSF integrating passive thermal mass in the air channel. The airflow inside the channel was assumed one-dimensional and vertical, so no airflow modelling was needed and the given airflow rate was directly applied to reckon the convective heat transfer coefficients. On the other hand, when the facade is naturally ventilated, the nodal unidirectional airflow network method is applied. The authors used the numerical model to compare the annual heating and cooling loads of the new system against traditional DSF with and without thermal mass walls. The results showed that the use of the new system can reduce the cooling load in a 27% in comparison to a traditional DSF. It was also highlighted that the use of any of the ventilated facades skin (inner or outer wall) as a mass component reduces the cooling load, but increases the energy consumption during the winter period. This is because the thermal mass increases the stack effect inside the cavity.

The natural ventilation and thermal performance of DSF have been deeply studied using the software package TAS for the thermal analysis of buildings. In the TAS program the airflow network model is coupled with an energy simulation algorithm which includes a 2D CFD package. Gratia and De Herde [33] analysed the parameters that influence the greenhouse effect (Fig. 3) in a DSF cavity (solar radiation level, orientation and shading devices, opaque walls proportion, wind

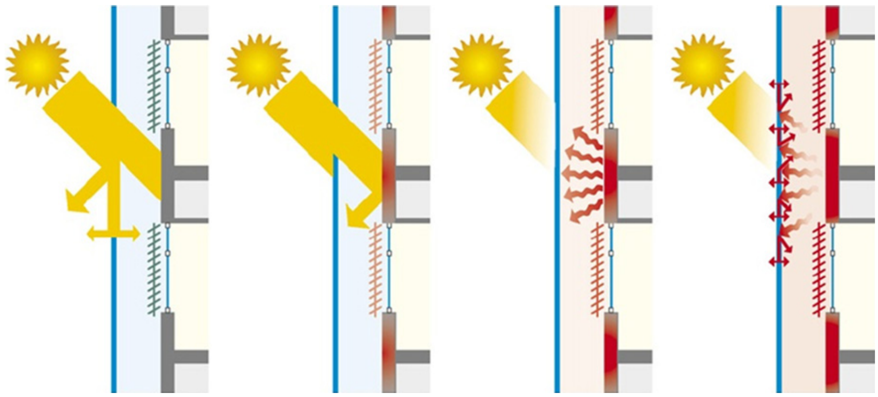

Fig. 3. Greenhouse effect in a DSF [33].

speed, colour of shading devices, depth of the cavity, glazing type, and openings). The authors concluded that the greenhouse effect must be minimized when no natural cooling strategies are used. On the other hand if those strategies are applied, the greenhouse effect is favourable when the DSF is south oriented. Similarly, the importance of the dynamic use of DSF was shown [34], as well as the crucial need of using control systems depending on the climatic conditions to program the use of shading devices and the day and night ventilation. Otherwise the DSF could worsen the performance of the whole building.

The position and colour of the shading devices used in a DSF of an office building were also studied using the TAS software [35]. Moreover, the energy demand during the heating and cooling periods of a building with and without DSF with different levels of insulations were compared [36], concluding that the orientation and use of heating and cooling natural strategies, the insulation level, and the internal gains have greater impact on the thermal performance of a building rather than the presence of a DSF. Hien et al. [37] used the TAS software to calculate the energy consumption, thermal comfort and condensations of single and double skin facade. The study demonstrated that under hot and humid weather conditions, naturally ventilated DSF are effective enough and the use of mechanical ventilation would provide negligible improvements. During the night, a fan must be employed to remove and prevent moisture condensation in the air channel.

The natural ventilation in office buildings with multi-storey DSF were studied by Gratia and De Herde [38–40] also using the TAS software. The authors provided guidelines related to the size and orientation of the openings in order to achieve a ventilation rate of 4 ACH under different wind conditions. Moreover, ventilation due to the stack effect and wind pressure was analysed to avoid overheating of the air gap. The authors concluded that cross-ventilation is less effective than the single-sided ventilation [40]. However it needs opening sizes 15–20 times smaller for similar air flow rates.

The building energy simulation program EnergyPlus has been also used to simulate the energy performance of DSF systems. The air cavity of the DSF can be divided as several zones and each zone is associated with an airflow network model. Chan et al. [41] analysed the optical and thermal impact of using different types of glasses in a DSF located in Hong Kong. These authors stated that the configuration of a DSF system with a single clear glass as an inner pane and a double reflective glazing as the outer pane is the most suitable solution under these environmental conditions. Even though a cooling energy demand reduction of 26% can be achieved, the high investment and maintenance costs of the DSF makes the system economically infeasible in Hong Kong. Furthermore, Hashemi et al. [42] validated a numerical model based on EnergyPlus using field measurements made on a hot arid climate during summer and winter periods for a DSF of an office high-rise building in Tehran. They concluded that night ventilation in the building is essential during summer to prevent increased cooling loads. From the simulation study it was demonstrated that both heating and cooling loads were reduced when a DSF is used.

5. Control volume approach

In the control volume approach, each skin of the DSF is divided into control volumes (approximately 1 m high), which are only coupled due to the presence of the air channel. This air channel is discretized orthogonally to the facade as shown on Fig. 4, so the mass flow rate of each control volume is equal to the mass flow rate at the inlet [9]. Moreover, the thermal gradient in the vertical direction is taken into account using this methodology. Faggembau et al. [43,44] implemented this approach using the AGLA code [45] to study numerically the thermal performance of a DSF and validated its algorithm with analytical solutions where possible and experimental measurements. The numerical model proved that the use of curtains inside the air channel instead of inside the building reduces the heat gains and that Low-ε coating reduces overheating during summer for facades with blinds inside the channel.

Following the same technique, Saelens et al. [46,47] studied the annual energy performance of an office building with different multiple-skin facades implementing the control volume method in TRNSYS (Fig. 5). The modelling environment consists of four models which are the facades, the office zone, the heating and cooling system, and the building energy management system. The energetic performance of three different multiple-skin facades, airflow window, DSF, and supply window (Fig. 6), and two traditional cladding systems, exterior and interior shading devices, were analysed numerically. The authors also highlighted the importance of using control strategies such as controlling the airflow rate and the recovery of air returning from the DSF in order to improve the energy efficiency of all facade systems. Moreover, Saelens et al. [48] demonstrated with experimental data and a sensitivity numerical study that the assumption of an inlet temperature equal to the exterior or interior air temperature is not valid. The inlet temperature must be estimated depending on the heating and cooling with the bounding surfaces and the heating due to solar radiation.

6. Zonal approach

The zonal approach modelling was developed by Jiru and Haghighat [49] and was applied to evaluate the thermal

performance of a DSF with venetian blinds. This methodology is an intermediate approach between the extremes of the lumped model and CFD, since DSF can be divided into a number of control volumes (2D or 3D), usually larger than the cells used in CFD models, so the system of algebraic equations to solve is smaller and much easier to solve in comparison to CFD methodology. An sketch of the zonal approach for a mechanically ventilated DSF with venetian blinds in the air channel is shown in Fig. 7.

The zonal model [49] was validated against experimental results and proves to provide information which is not possible for the lumped and the control volume approach models, moreover the described method does not need the computational requirements as CFD does. Finally, the model is used to study how the temperature difference between inlet and outlet varies depending on the inlet flow rate, the height of the DSF, and the presence or absence of the venetian blinds.

7. Numerical solution of partial differential equations and computational fluid dynamics (CFD)

The numerical methods for solving partial differential equations (PDEs) are based on replacing the differential equations by algebraic equations. The three classical choices for the numerical solution of PDEs are the finite difference method (FDM), the finite element method (FEM) and the finite volume method (FVM).

In the case of the popular finite difference method, this is done by replacing the differential quantities by sufficiently small differences [50]. Akbari and Borgers used the fully implicit finite difference model to study the free convective laminar [51] and turbulent [52] flows between parallel vertical plates. Empirical correlations were obtained from the model to predict the induced flow rate, the total height and the heat transfer rate between walls and fluid. Moreover, the convective laminar heat transfer between

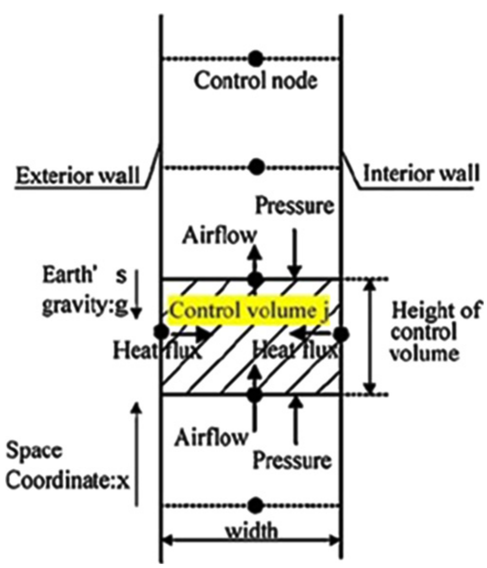

Fig. 4. Control volume discretization of the air channel of a DSF system [43].

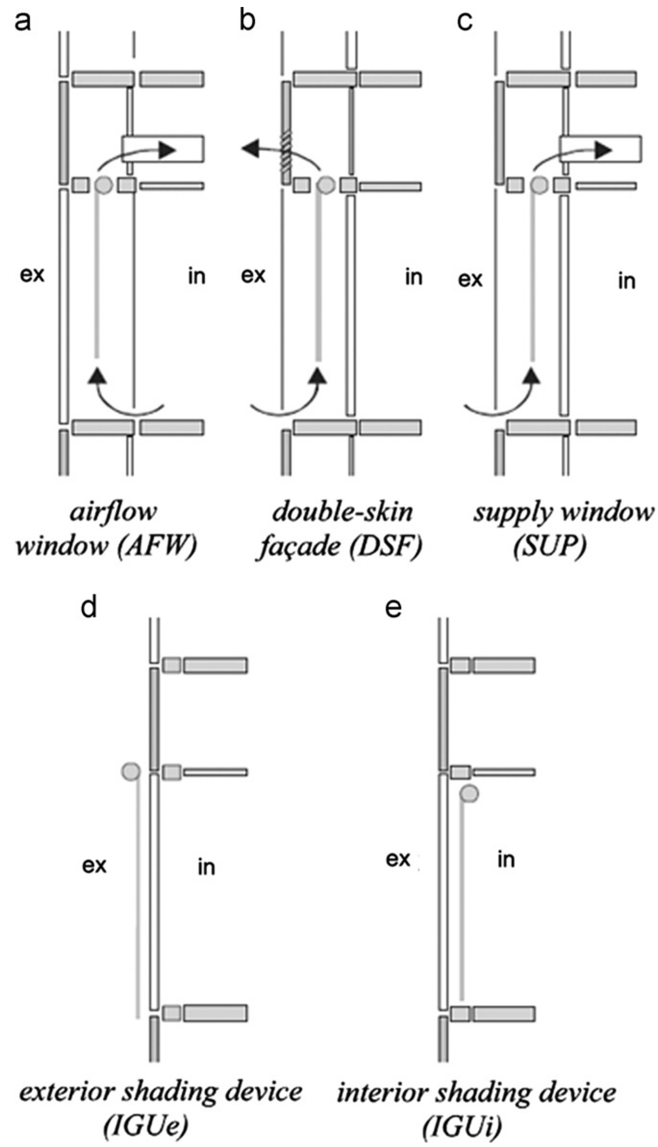

Fig. 6. Schematic representation of the multiple skin facades and the traditional solutions [47].

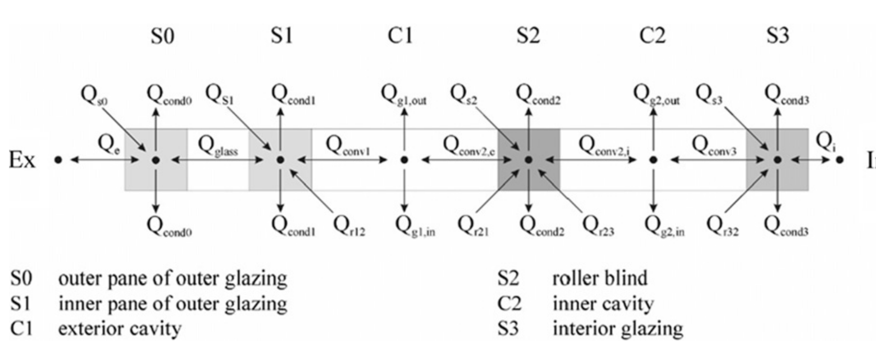

Fig. 5. Control volume discretization of a ventilated facade with lowered roller blind (Q is the heat flux; Q_s is the absorbed solar energy; Q_c is the convective heat transfer; Q_{cond} is the conductive heat transfer; Q_r is the radiation heat transfer) [46].

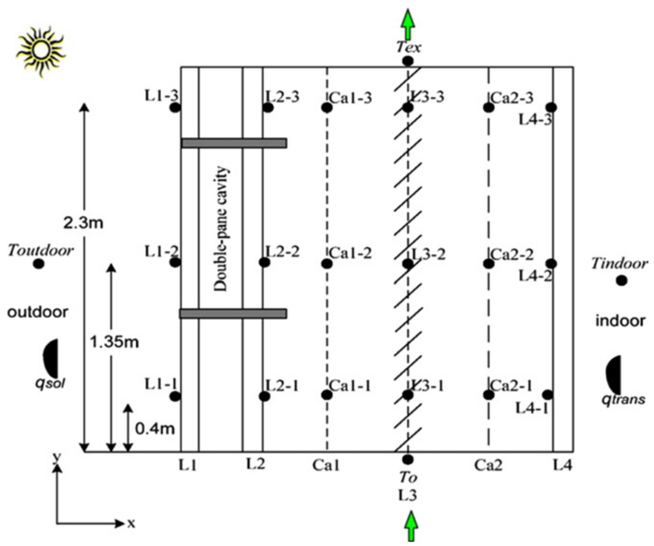

Fig. 7. Mechanically ventilated DSF: L1 is the exterior glass of the double-pane; L2 is the interior glass of the double-pane; L3 is the venetian blinds; L4 is the interior glass of the ventilated DSF; Ca1 is the outer cavity; and Ca2 is the inner cavity, T_0 is the air temperature at the inlet, T_{ex} is the air temperature at the exit, T_{indoor} is the room air temperature, $T_{outdoor}$ the outside air temperature, q_{sol} is the total solar radiation, q_{trans} is the transmitted solar radiation, ● is the thermocouple and ◖ is the pyranometer [49].

the air channel of a Trombe wall and its surfaces have been numerically investigated by Jubran et al. [53]. The numerical analysis included a conventional Trombe wall and a modified one with a tilt angle in the outer glass wall, concluding that the modified version gives better thermal performance.

Aside from using this methodology to determine the convective heat transfer coefficient, the finite differences method has been used to compare different types of DSF. Shen et al. [54] studied numerically the thermal performance of a classical and a composite Trombe wall. The authors developed a simulation model based on finite differences method and compared the results to the type 36 of TRNSYS software [55]. The model presented better agreement with experimental data rather than the type 36 since it does not assume constant discharge coefficients. In addition, Zalewski et al. [56] developed a one-dimensional model based on finite differences to describe the thermal behaviour of four different types of solar walls. The model was validated using experimental data and was used to compare the efficiency in solar heat storage of classical, composite, and insulated Trombe walls, and non-ventilated solar walls. Moreover, the supply in summertime of the different solar walls was also numerically compared. It was shown that the classical Trombe wall is the constructive system that maximizes the heat absorption, however it is difficult to control the heat supplied during summer which may produce overheating. On the other hand, the use of composite solar walls reduces drastically this possible overheating during summer periods. Furthermore, a factorial plan was carried out in the paper to conclude that to maximize efficiency of a composite solar wall, the storage wall should not be very thick and its absorption coefficient should be as high as possible.

In addition, the heat losses of a single and double skin facade office buildings were compared during winter in Istanbul [57]. A numerical model based on two steps (inter-space temperature and finite difference method) was used to point out that the use of DSF would reduce in a 40% the heat losses during winter.

Finally the thermal modelling of DSF with integrated photovoltaics (PV) has been addressed using finite difference method. Mei et al. [58] developed a numerical model to analyse an integrated ventilated PV facade. This model was implemented with TRNSYS building model, and was validated experimentally with measurements of a building in free floating conditions. Moreover, Charron and Athienitis [59] studied different configurations and the use of fins. It was concluded that even though the use of PV modules inside a cavity of a DSF reduces the electricity generation of the PV in a 21% due to the outer skin reflects part of the solar radiation, its efficiency increases about a 25% because of the heat rejected by the fans in the cavity.

Moreover, the finite element method was used by Agnoletto et al. [60] to solve the coupled conduction, convection, and long-wave radiation problem in a building envelope with DSF. Both inside and outside ambient temperatures were assumed constant, while the solar radiation varied in a sinusoidal form. The model determined the temperature distribution at each relevant surface, the air temperature and velocity distribution in the channel, the heat fluxes and the solar transmitted energy to the inside.

The finite volume method was used by Fedorov and Viskanta [61] to model the convective heat transfer coefficients in an asymmetrically heated vertical channel due to turbulent natural flow. The authors used a low Reynolds number k–ε turbulence model, which was experimentally validated and used to provide empirical correlations.

The finite volume method was also used to study numerically the DSF by many researchers. This method, developed by Patankar [62], discretizes the governing equations (Navier–Stokes and energy conservation) dividing the physical space into a finite number of control volumes. Yedder et al. [63] analysed numerically the natural convection in a Trombe wall solar collector using a constant heat flux boundary condition at the solar radiation receiving surface. After model validation it was concluded that the aspect ratio has a small influence on the heat transfer, while other parameters such as orifice position, channel width and size, have high influence in the thermal performance of the envelope. Mootz and Bezian [64] studied the influence of channel depth in the convective heat transfer. Two different options were introduced to solve the channel depth variation, changing or maintaining the total thickness of the wall, hence varying or not the insulation thickness. The results extracted from this numerical study concluded that if the thickness of the wall is maintained, the best panel performance during recovery periods requires a maximum channel spacing, while this channel must be minimized during non-recovery periods, since less insulation is used and hence there are more heat losses to the environment. On the other hand, large channel spacing proved to be the most efficient in both recovery and non-recovery periods when the thickness of the wall can be modified. Moreover, Balocco [65] presented a simple steady model to demonstrate that natural convection effect is affected by the ratio between channel width and height, since it affects the stack effect and the wall frictional resistance.

Hossegen et al. [66] and Seferis et al. [67] used the transient simulation program ESP-r, which is based on the finite volume technique, to study numerically the performance of DSF. Hossegen et al. [66] used this commercial software to study whether a double-skin should be applied to the east facade of a building in Norway. They concluded that the use of DSF reduces the heating energy demand in a 20%. However, it is not interesting from an economical point of view since the electrical energy savings will not defend the additional costs the DSF constitute. Furthermore, Seferis et al. [67] validated experimentally the model showing good agreement except for night periods during summer. The authors used the model to point out the improvements in the thermal behaviour of the facade due to the addition of a radiant barrier layer.

The CFD technique has been widely used for air flow simulation in the channel of a ventilated DSF. The air flow model consisted of a system of governing equation representing continuity, momentum,

turbulence, enthalpy, and concentration [68] solved for all nodes of a two-or three-dimensional grid in order to provide detailed information about the nature flow field. This numerical approach uses weather the FDM or the FVM to solve these equations. Hensen et al. [25] stated that even though the CFD approach can be applied to any thermofluid investigation, in the building physics domain, there are several problems to address, such as the computing power, the nature of the flow fields, the assessment of the complex, and the occupant-dependent boundary conditions. These limitations led to CFD studies being restricted to steady-state cases, very short simulation periods or the study of the convective heat transfer across the DSF surfaces and provide empirical correlations [69–72], where the aspect ratio of each node must be small enough so the grid can capture boundary layer effects [9].

Pasut and De Clari [73] have discussed which factors are important in the simulation of a naturally ventilated facade using the CFD technique, and which factors can be neglected since they only increase the computational cost of the simulation. A sensitivity analysis shows that the thermal conductivity and heat capacity of the air could be treated as a constant, since their values are calculated as functions of temperature, they only increase the execution time without providing significant improvement in the results. Moreover k-ε RNG turbulent model is suggested in comparison to k–ω models, due to its better agreement with experimental data. The authors have also highlighted that for natural ventilated DSF the velocity fieldd is almost bidirectional, hence 3D models are not needed.

Xamán et al. [70] used a two-dimensional steady state numerical model to describe the natural convection effect of a tall rectangular cavity in laminar and turbulent regimes (air channel of a DSF). Four different turbulence models k–ε were compared against experimental data, demonstrating that the model proposed by Ince and Launder [74] predicted the measured results more accurately. Empirical correlations of overall convective Nusselt numbers are given for different aspect ratios [68]. Moreover, Coussirat et al. [75] analysed different models of turbulence and radiation to simulate the thermal performance of a double glazed facade using CFD. The model was validated against Manz experimental data [76], and the study concluded that for this configuration P-1 Radiation model, and Standard and RNG k–ε model showed the smallest error in all cases. A mesh sensitivity test was also carried out, observing a clear dependence on the mesh density.

Validation of the numerical results using experimental data or analytical solutions must be done whatever which typology of modelling is used to describe the thermal performance of the DSF system. However, since CFD technique allows to calculate temperature, pressure and velocity maps, there are few experimental data which can be used to validate CFD models. Ding et al. [77] used reduced scale model experiments and CFD analysis to study the natural ventilation of a DSF with a thermal storage space called solar chimney. The experimental model was reproduced as 1/25 of the full-scale building. Here a solar chimney was used to strength the stack effect, ensuring natural convection and minimizing the effect of wind. The area of the openings was optimized depending on the airflow and pressure differences. Moreover, Gosselin et al. [78] presented a new computational method to describe the heat transfer and air flow in a dual-airflow window. The method consists of four steps and was validated with experimental tests on a full-scale dual-airflow window system. First, CFD is used without including radiation in the model, second a separated code calculates surface to surface radiation and solar radiation. The output from this second step is introduced in another CFD simulation as heat sources or sinks. Finally an average between temperature profiles of CFD models with and without radiations is done.

Even though some authors have developed their own CFD code to study the thermal performance of the DSF systems [79], commercial CFD packages have been widely used. An spectral optical model combined with CFD was implemented using Flovent to analyse the thermal behaviour of a DSF [76]. The model was validated with experimental data, extracted from a test element provided with approximately 60 sensors. It was concluded that the sequence alteration of a given set of layers in a glass double facade or the ventilation properties of the facade can vary drastically the solar energy transmittance. The same model was used by Manz et al. [80] to show that the change in the orientation of the forced flow relative to the gravitational field, influences the total solar energy gain, and hence cannot be modelled using a piston-flow model, but using CFD. Moreover Pérez-Grande et al. [81] used Fluent CFD package to study the effect of the glass properties on the thermal performance of a double-glazed facade. Total heat flux gains through the facades with different combinations of outer and inner glasses were compared. Naturally and mechanically ventilated facades were also evaluated using the model concluding that the selection of the glasses is a key point in the design of the double glassed facades, since the thermal loads into the buildings can vary an order of magnitude. Furthermore, Patania et al. [82] compared the thermal performance of three different opaque ventilated facades using Fluent CFD package. The parametric study included the analysis of different solar radiation intensity, inlet velocity and temperature. This study showed that the electrical energy savings during summer periods of an opaque ventilated facade can achieve 40%, and that the most important thermo-physic characteristics that affects the behaviour of the system is the thermal diffusivity of the external layer, standing that the lower is the diffusivity, the better energy performance of the ventilated facade. It was also remarked that the velocity profiles along the cavity are symmetric and two boundary layers are developed along both sides. Guardo et al. [83] also used Fluent package to study which parameters affect the most the reduction of solar load gain in a DSF. The numerical study concluded that the optical properties of the glass have a critical importance in the design of DSF, replacing the internal glazing for a low-emissivity glass reduces the solar load by half. Meanwhile a reduction of 55% of the transmissivity of the external glazing can provide 40% of solar load reductions. Finally, Wong et al. [84] used the Fluent CFD solver engine to evaluate the impact of using DSF in a high rise office building of 18 storeys. Their results indicate that DSF can be used to introduce natural ventilation to the high-rise buildings in the tropics.

As it was previously said, CFD simulation can provide information about the nature flow field, this is why this approach is the only way to solve some details in the design of a DSF, such as flow around venetian blinds [85–87], openings [88], and different shading systems (Fig. 8) [89]. Moreover, Safer et al. [86] assessed the modelling of a forced ventilated DSF with a venetian blind. 2D and 3D models were carried out to parameterize the air outlet position, the slat tilt angle, and the position of the blinds. Here, an homogeneous porous media model was used to reduce the number of meshes. The authors concluded that the slat tilt angle is important when the blind is in the centre of the channel and the facade is externally ventilated. Furthermore, for cases of external ventilation, it was found that the blinds have to be placed close to the inner surface to maximize the air flow and hence the heat transfer coefficients. Xu and Yang [87] coupled an optical model, an interface heat balance model, and the CFD model to study the natural ventilation in a DSF with venetian blinds. The numerical results were validated using the experimental data from Manz [75]. Moreover, Sanjuan et al. [88] developed a 3D CFD model to analyse the fluid dynamics and thermal performance of an open joint ventilated facade (Fig. 9), and to compare its behaviour against a conventional sealed ventilated facade. The results pointed out that while in a conventional ventilated facade the natural convection effect produces a loop in the air gap (where the air near the heated

surface ascends and the air near the cooled descends), in the open joint ventilated facade, the air can come in and out in every opening, which reduces the convective flow. In this study it was concluded that the use of an open joint ventilated facade lowered the temperatures in the air gap, producing a reduction of 26% in the

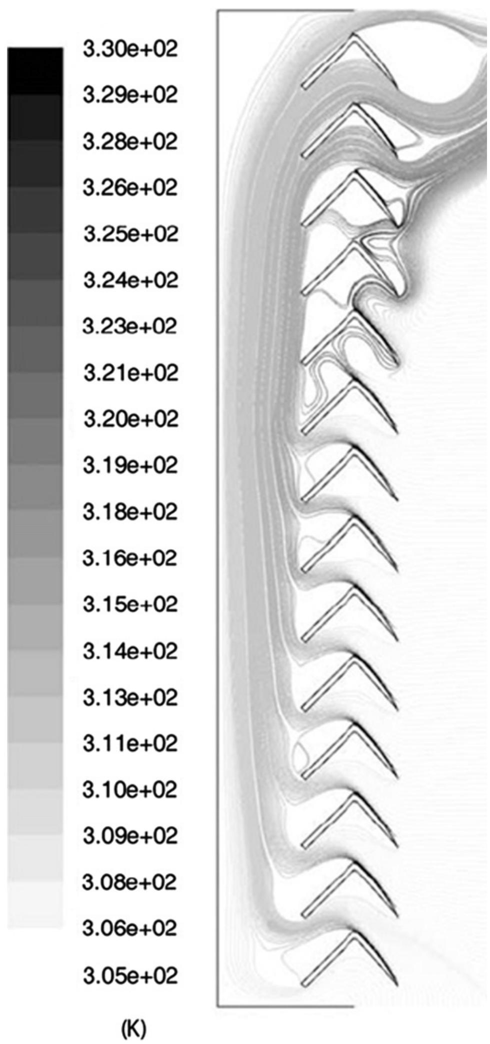

Fig. 8. Temperature air streamlines (K) around the shading system proposed by Baldinelli [89].

heat gains during the summer period. However, this temperature reductions lead to increase the heat losses 50% during winter time.

Even though CFD can provide useful information about the airflow occurring in the DSF, the complete thermal and airflow description of these systems requires a coupled model including the modelling of optics (spectral method), the thermodynamics and fluid dynamics of the air chamber and room space (CFD), and a building energy simulation tool. An important effort has been done

Manz and Frank [90] developed a coupled model which is economical in terms of computing time. The model was used to highlight the natural night-time ventilation. Furthermore, Baldinelli [89] studied numerically a glass DSF with a movable shading device using an optical, CFD, and building energy models. Simulations under 2D approaches were validated with experimental data showing that the use of 2D instead of 3D approach give an almost negligible additional error. The model was used to compare the studied DSF against traditional constructive systems in the north of Italy such as glassed and opaque facades, showing significant improvements in the building energy behaviour (reducing the heating load in winter and minimizing the cooling load in summer) especially when forced convection is used in the air chamber.

8. Integration between building energy and airflow models

The integration of building energy simulation (ES) and airflow model, usually CFD, can provide complementary information of the building performance. Moreover, the coupling simulation could better enhance the boundary conditions assumed for both, the thermal and airflow models. This coupled modelling is not only interesting for the numerical simulation of DSF but for any thermal zone of the building, as well.

According to Srebric et al. [91] there are three main discontinuities between ES and CFD: the time-scale, the modelling, and the speed discontinuity. The first discontinuity is produced because ES has a characteristic time-scale of hours while CFD has a few seconds. The modelling discontinuity appears since the air temperature is spatially averaged in the ES models and represented as a field distribution in CFD. In addition, the speed discontinuity reflexes the differences in the computing time needed for each model. Zhai et al. [92] describe some efficient approaches to bridge these discontinuities, such as static and dynamic coupling strategies. The static coupling involves one-step or two-step exchange of information between ES and CFD as seen in Fig. 10. The static coupling is appropriate in the cases where ES, CFD, or both are not very sensitive to the exchanged variables such as convective heat transfer

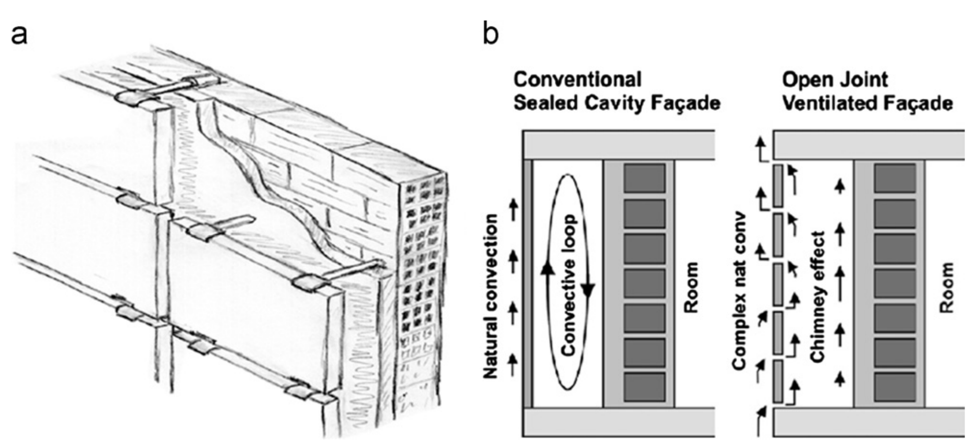

Fig. 9. (a) Sketch of an open-joint ventilated façade. (b) Differences in heat transfer processes between a conventional sealed cavity facade and an open joint ventilated facade. [88].

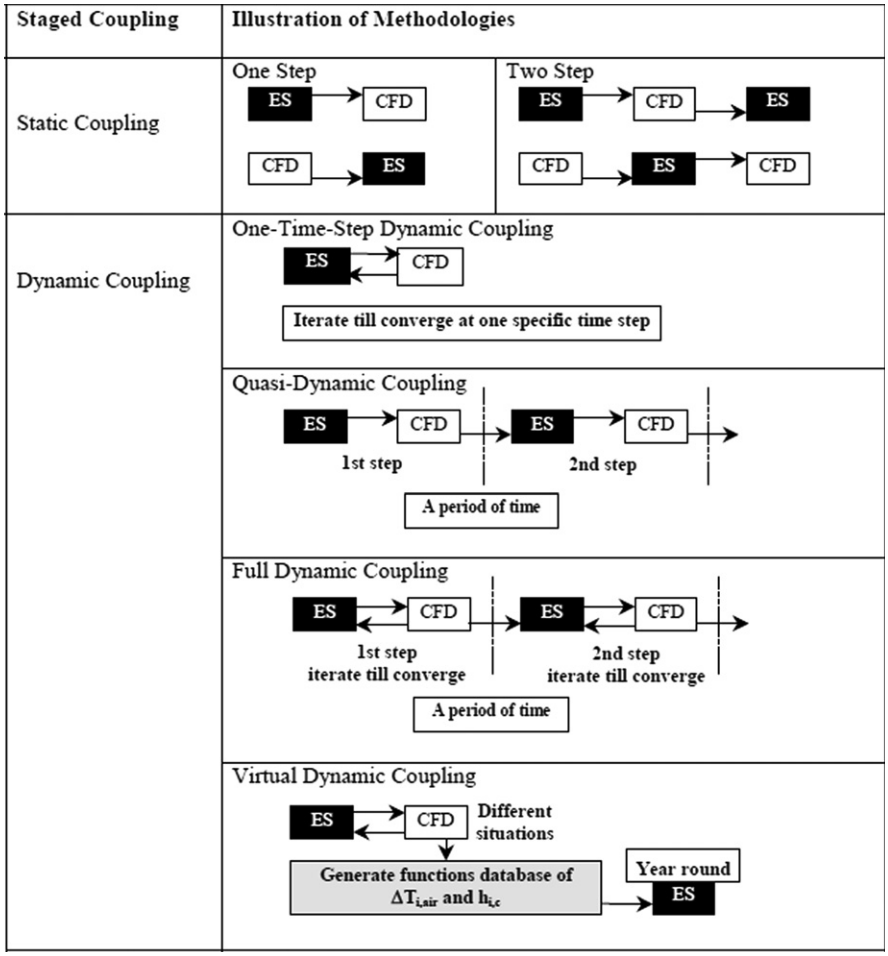

Fig. 10. Illustration of the staged coupling strategies (the arrow from CFD to ES indicates the transfer of thermal air distribution and convective heat transfer coefficient, while the arrow from ES to CFD indicates the transfer of surface temperatures and $Q_{heat\_extraction}$).

coefficients or thermal air distributions. On the other hand, dynamic coupling involves coupling between the two models at every time-step. In the one-time-step dynamic coupling and the full dynamic coupling, the iteration between ES and CFD is performed until convergence, which supposes a higher computational cost. Quasi-dynamic coupling requires less computational effort and is advised when the time-step is small and virtual dynamic coupling, as proposed by Chen and van der Kooi [93] is the most suitable coupling method for a whole year energy analysis.

Zhai and Chen [94] have developed a coupled program (EnergyPlus and MIT-CFD) incorporating different coupling strategies and the numerical results are validated in four different experimental facilities. The authors conclude that even though the coupled simulation took much longer computing time than energy simulation alone, the numerical results are more accurate, especially in the calculation and assumption of convective heat transfer coefficients.

9. Conclusions

The use of double skin facades (DSF) in the building sector has become popular and has been widely studied. Computer numerical simulation is one of the most powerful techniques in the design process of these systems. This paper overviews the existing numerical methods used to predict the thermal performance of DSF under different environmental conditions. The main conclusions which can be extracted from this study are summarised as follows:

- Several hypotheses must be assumed to solve the analytical and lumped models. However, they can generally provide specific useful information without consuming high computational resources.
- Dimensionless analysis is presented as a basic tool to calculate heat transfer through the walls of a DSF without using simulation programs of different complexity levels. Even though the dimensionless studies presented in this paper are based on experimental measurements, numerical tools, such as CFD, can also be useful to develop dimensionless correlations which describe the thermal performance of the system.
- The airflow network model can provide fast useful information about bulk flows without consuming high computational resources, and is usually integrated with a thermal network, and a building energy model.
- The control volume approach is based on one-dimensional discretization (in the indoor–outdoor direction for the solid layers, and in the flow direction for the air channel). This approach provides a good compromise between computational resources and accuracy.
- The zonal model has proved to provide information which is not possible for the lumped and the control-volume models.
- Computational fluid dynamics simulation is the unique way to solve some details in the design of a DSF, such as flow around venetian blinds, openings, and different shading systems.

Its use in the building sector is limited because of problems related to the computing power, the nature of the flow fields, the assessment of the complex, and the occupant-dependent boundary conditions. When using this numerical approach to model a ventilated facade, k–ε RNG turbulence model is recommended, and 3D models are not necessary since velocity field is almost bidirectional.
- The integration of building energy simulation (ES) and airflow model using CFD can provide more accurate prediction to study the thermal performance of a DSF or a whole building system.
- The different discontinuities between the ES and the CFD can be addressed using different coupling strategies, with different computational costs. Virtual dynamic coupling is the most suitable coupling method for a whole year energy analysis, while the static coupling is recommended in the cases where ES, CFD, or both are not very sensitive to the exchanged variables such as convective heat transfer coefficients or thermal air distributions.
- CFD simulation might be used to build numerical correlations which describe some specific parameters such as convective heat transfer coefficients. These correlations might be used by engineers or architects in simplified models during the design phase of the DSF.

Most of the models presented were validated against experimental results. However, further research is required to compare all the models with the same experimental test in order to evaluate accuracy and consumption of computational resources. Moreover, the researchers of this study want to highlight the potential and the importance of developing coupled models, which are able to use detailed information from CFD in an overall energy simulation model describing the thermal performance of a whole building.

Acknowledgements

The work was partially funded by the Spanish government (ENE2011-28269-C03-01) and the European Union (COST Action COST TU0802). The authors would like to thank the Catalan Government for the quality accreditation given to their research group (2009 SGR 534).

References

[1] Directive 2010/31/eu of the European parliament and of the council of 19 May 2010 on the energy performance of buildings; 2010. Available from: 〈http://www.epbd-ca.eu〉.
[2] Cabeza LF, Castell A, Medrano M, Martorell I, Pérez G, Fernández I. Experimental study on the performance of insulation materials in Mediterranean construction. Energy and Buildings 2010;42:630–6.
[3] Shameri MA, Alghoul MA, Sopian K, Fauzi M, Zain M, Elayeb O. Perspectives of double skin façade systems in buildings and energy saving. Renewable and Sustainable Energy Reviews 2011;15:1468–75.
[4] Safer N, Woloszyn M, Roux JJ. Three-dimensional simulation with a CFD tool of the airflow phenomena in single floor double-skin façade equipped with a Venetian blind. Solar Energy 2005;79:193–203.
[5] Ding W, Hasemi Y, Yamada T. Natural ventilation performance of a double-skin façade with a solar chimney. Energy and Buildings 2005;37:411–8.
[6] Wang PC. Natural ventilation in double-skin façade design for office buildings in hot and humid climate. PhD thesis. University of New South Wales, Australia, December; 2008.
[7] Faggembauu D. Heat transfer and fluid-dynamics in double and single skin facades. PhD thesis. Universitat Politècnica de Catalunya, Spain, December; 2006.
[8] Poirazis H. Double skin façades for office buildings. Report EBD-R-04/3. Lund University; 2004. T.
[9] Zhou J, Chen Y. A review on applying ventilated double-skin façade to buildings in hot-summer and cold-winter zone in China. Renewable and Sustainable Energy Reviews 2010;14:1321–8.
[10] Zalewski L, Chantant M, Lassue S, Duthoit B. Experimental thermal study of a solar wall of composite type. Energy and Buildings 1997;25:7–18.
[11] Torcellini P, Pless S. Trombe walls in low-energy buildings: practical experiences. In: VII World renewable energy congress, Denver; 2004.
[12] Pasquay T. Natural ventilation in high-rise buildings with double facades, saving or waste of energy. Energy and Buildings 2004;36:381–9.
[13] Gavan V, Woloszyn M, Kuznik F, Roux JJ. Experimental study of a mechanically ventilated double-skin façade with venetian sun-shading device: a full investigation in controlled environment. Solar Energy 2010;84:183–95.
[14] Huckermann V, Kuchen E, Leão M, Leão É. Empirical thermal comfort evaluation of single and double skin façades. Building and Environment 2010;45:976–82.
[15] Khadir Nasri. A model of a Trombe wall. Renewable Energy 1991;1:533–41.
[16] Holmes MJ. Optimization of the thermal performance of mechanically and naturally ventilated glazed facades. Renewable Energy 1994;5:1091–8.
[17] von Grabe J. A prediction tool for the temperature field of double facades. Energy and Buildings 2002;34:891–9.
[18] Ciampi M, Leccese F, Tuoni G. Ventilated facades energy performance in summer cooling of buildings. Solar Energy 2003;75:491–502.
[19] Ong KS. A mathematical model of a solar chimney. Renewable Energy 2003;28:1047–60.
[20] Hirunlabh J, Kongduang W, Namprakai P, Khedari J. Study of natural ventilation of houses by a metallic solar wall under tropical climate. Renewable Energy 1999;18:109–19.
[21] Park C, Augenbroe G, Messadi T, Thitisawat M, Sadegh N. Calibration of a lumped simulation model for double-skin façade systems. Energy and Buildings 2004;36:1117–30.
[22] Park C, Augenbroe G, Sadegh N, Thitisawat M, Messadi T. Real-time optimization of a double-skin façade based on lumped modeling and occupant preference. Energy and Buildings 2004;39:939–48.
[23] Balocco C. A non-dimensional analysis of a ventilated double façade energy performance. Energy and Buildings 2004;36:35–40.
[24] Balocco C, Colombari M. Thermal behavior of interactive mechanically ventilated double glazed façade: non-dimensional analysis. Energy and Buildings 2006;38:1–7.
[25] Hensen J, Bartak M, Drkal F. Modeling and simulation of a double-skin façade system. ASHRAE Transactions 2002;108:1251–9.
[26] Aynsley R. A resistance approach to analysis of natural ventilation airflow networks. Journal of Wind Engineering and Industrial Aerodynamics 1997;67&68:711–9.
[27] Tanimoto J, Kimura K. Simulation study on an air flow window system with an integrated roll screen. Energy and Buildings 1997;26:317–25.
[28] Stec W, Van Paassen D. Defining the performance of the double skin façade with the use of the simulation model. In: Eighth international IBPSA conference, Eindhoven; 2003.
[29] Stec W, Van Paassen D. Symbiosis of the double skin façade with the HVAC system. Energy and Buildings 2005;37:461–9.
[30] Stec W, Van Paassen D, Maziarz A. Modelling the double skin façade with plants. Energy and Buildings 2005;37:419–27.
[31] Haase M, Marques da Silva F, Amato A. Simulation of ventilated facades in hot and humid climates. Energy and Buildings 2009;41:361–73.
[32] Fallahi A, Haghighat F, Elsadi H. Energy performance assessment of double-skin façade with thermal mass. Energy and Buildings 2010;42:1499–509.
[33] Gratia E, De Herde A. Greenhouse effect in doublé-skin facade. Energy and Buildings 2007;39:199–211.
[34] Gratia E, De Herde A. Optimal operation of a south double-skin facade. Energy and Buildings 2004;36:41–60.
[35] Gratia E, De Herde A. The most efficient position of shading devices in a double-skin facade. Energy and Buildings 2007;39:364–73.
[36] Gratia E, De Herde A. Are energy consumptions decreased with the addition of a double-skin. Energy and Buildings 2007;39:605–19.
[37] Hien WN, Liping W, Chandra AN, Pandey AR, Xiaolin W. Effects of double glazed façade on energy consumption, thermal comfort and condensation for a typical office building in Singapore. Energy and Buildings 2005;37:563–72.
[38] Gratia E, De Herde A. Guidelines for improving natural daytime ventilation in an office building with a double-skin facade. Solar Energy 2007;81:435–48.
[39] Gratia E, De Herde A. Natural ventilation in a double-skin facade. Energy and Buildings 2004;36:137–46.
[40] Gratia E, De Herde A. Is day natural ventilation still possible in office buildings with a double-skin façade? Building and Environment 2004;39:399–409.
[41] Chan ALS, Chow TT, Fong KF, Lin Z. Investigation on energy performance of double skin façade in Hong Kong. Energy and Buildings 2009;41:1135–42.
[42] Hashemi N, Fayaz R, Sarshar M. Thermal behavior of a ventilated double skin façade in hot arid climate. Energy and Buildings 2012;42:1823–32.
[43] Faggembau D, Costa M, Soria M, Oliva A. Numerical analysis of the thermal behaviour of ventilated glazed facades in Mediterranean climates. Part I: Development and validation of a numerical model. Solar Energy 2003;75:217–28.
[44] Faggembau D, Costa M, Soria M, Oliva A. Numerical analysis of the thermal behaviour of ventilated glazed facades in Mediterranean climates. Part II: Applications and analysis of results. Solar Energy 2003;75:229–39.
[45] Costa M, Acebes O, Sen F, Platzer W, Haller A, Indetzki M, et al. Analysis of multifunctional ventilated facades. An European Joule Project. In: Proceedings of Eurosun conference, Copenhagen; 2000.
[46] Saelens D, Carmeliet J, Hens H. Energy performance assessment of multiple-skin facades. HVAC&R Research 2003;9:167–85.
[47] Saelens D, Roels S, Hens H. Strategies to improve the energy performance of multiple-skin facades. Building and Environment 2008;43:638–50.

[48] Saelens D, Roels S, Hens H. The inlet temperature as a boundary condition for multiple-skin facade modelling. Energy and Buildings 2004;36:825–35.
[49] Jiru TE, Haghighat F. Modeling ventilated double skin façade—A zonal approach. Energy and Buildings 2008;40:1567–76.
[50] Çengel Y. Heat transfer. A practical approach. McGraw-Hill; 1998.
[51] Akbari H, Borgers TR. Free convective laminar flow within the Trombe wall channel. Solar Energy 1979;22:165–74.
[52] Borgers TR, Akbari H. Free convective turbulent flow within the trombe wall channel. Solar Energy 1984;33:253–64.
[53] Jubran BA, Hamdan MA, Manfalouti W. Modelling free convection in a Trombe wall. Renewable Energy 1991;1:351–60.
[54] Shen J, Lassue S, Zalewski L, Huang D. Numerical study on thermal behavior of classical or composite Trombe solar walls. Energy and Buildings 2007;39:962–74.
[55] Solar Laboratory of energy (USA), Manuals of TRNSYS, University of Wisconsin-Madison, USA; 1994.
[56] Zalewski L, Lassue S, Duthoit B, Butez M. Study of solar walls—validating a simulation model. Building and Environment 2002;37:109–21.
[57] Yilmaz Z, Çetintas F. Double skin façade's effects on heat losses of office buildings in Istanbul. Energy and Buildings 2005;37:691–7.
[58] Mei Li, Infield D, Eicker U, Fux V. Thermal modelling of a building with an integrated ventilated PV façade. Energy and Buildings 2003;35:605–17.
[59] Charron R, Athienitis AK. Optimization of the performance of double-façades with integrated photovoltaic panels and motorized blinds. Solar Energy 2006;80:482–91.
[60] Agnoletto L, Cortella G, Manzan M. Finite element termal analysis of special building components. Energy and Buildings 1995;22:115–23.
[61] Fedorov AG, Viskanta R. Turbulent natural convection heat transfer in an asymmetrically heated, vertical parallel-plate channel. International Journal of Heat and Mass Transfer 1997;40:3849–60.
[62] Patankar S. Numerical heat transfer. Hempisphere Publications; 1980.
[63] Yedder RB, Du ZG, Bilgen E. Numerical study of laminar natural convection in composite Trombe wall systems. Solar & Wind Technology 1990;7:675–83.
[64] Mootz F, Bezian JJ. Numerical study of a ventilated facade panel. Solar Energy 1996;57:29–36.
[65] Balocco C. A simple model to study ventilated facades energy performance. Energy and Buildings 2002;34:469–75.
[66] Hosseggen R, Wachenfeldt BB, Hanssen SO. Building simulation as an assisting tool in decision making Case study: With or without a double-skin façade? Energy and Buildings 2008;40:821–7.
[67] Seferis P, Strachan P, Dimoudi A, Androutsopoulos A. Investigation of the performance of a ventilated wall. Energy and Buildings 2001;43:2167–78.
[68] Gan Guohui. A parametric study of Trombe walls for passive cooling of buildings. Energy and Buildings 1998;27:37–43.
[69] Manz Heinrich. Numerical simulation of heat transfer by natural convection in cavities of facade elements. Energy and Buildings 2003;35:305–11.
[70] Xamán J, Álvarez G, Lira L, Estrada C. Numerical study of heat transfer by laminar and turbulent natural convection in tall cavities of façade elements. Energy and Buildings 2005;37:787–94.
[71] Yilmaz T, Fraser S. Turbulent natural convection in a vertical parallel-plate channel with asymmetric heating. International Journal of Heat and Mass Transfer 2007;50:2612–23.
[72] Pappas A, Zhai Z. Numerical investigation on thermal performance and correlations of double skin façade with buoyancy-driven airflow. Energy and Buildings 2008;40:466–75.
[73] Pasut W, De Carli M. Evaluation of various CFD modelling strategies in predicting airflow and temperature in a naturally ventilated doublé skin façade. Applied Thermal Engineering 2012;37:267–74.
[74] Ince N, Launder B. On the computation of buoyancy-driven turbulent flows in rectangular enclosures. International Journal of Heat Fluid Flow 1989;10:110–7.
[75] Coussirat M, Guardo A, Jou E, Egusquiza E, Cuerva E, Alavedra P. Performance and influence of numerical sub-models on the CFD simulation of free and forced convection in double-glazed ventilated façades. Energy and Buildings 2008;40:1781–9.
[76] Manz H. Total solar energy transmittance of glass double façades with free convection. Energy and Buildings 2004;36:127–36.
[77] Ding W, Hasemi Y, Yamada T. Natural ventilation performance of a doublé-skin façade with a solar chimney. Energy and Buildings 2005;37:411–8.
[78] Gosselin JR, Chen Q. A computational method for calculating heat transfer and airflow through a dual-airflow window. Energy and Buildings 2008;40:452–8.
[79] Onishi J, Soeda H, Mizuno M. Numerical study on a low energy architecture based upon distributed heat storage system. Renewable Energy 2001;22:61–6.
[80] Manz H, Schaelin A, Simmler H. Airflow patterns and thermal behavior of mechanically ventilated glass double façades. Building and Environment 2004;39:1023–33.
[81] Pérez-Grande I, Meseguer J, Alonso G. Influence of glass properties on the performance of double-glazed facades. Applied Thermal Engineering 2005;25:3163–75.
[82] Patania F, Gagliano A, Nocera F, Ferlito A, Galesi A. Thermofluid-dynamic analysis of ventilated facades. Energy and Buildings 2010;42:1148–55.
[83] Guardo A, Coussirat M, Egusquiza E, Alavedra P, Castilla RA. CFD approach to evaluate the influence of construction and operation parameters on the performance of active transparent façades in Mediterranean climates. Energy and Buildings 2009;41:534–42.
[84] Wong PC, Prasad D, Behnia M. A new type of double-skin façade configuration for the hot and humid climate. Energy and Buildings 2008;40:1941–5.
[85] Jiru TE, Tau Y, Haghighat F. Airflow and heat transfer in double skin facades. Energy and Buildings 2011;43:2760–6.
[86] Safer N, Woloszyn M, Roux JJ. Three-dimensional simulation with a CFS tool of the airflow phenomena in single floor double-skin facade equipped with a venetian blind. Solar Energy 2005;79:193–203.
[87] Xu X, Yang Z. Natural ventilation in the double skin facade with venetian blind. Energy and Buildings 2008;40:1498–504.
[88] Sanjuan C, Suárez MJ, González M, Pistono J, Blanco E. Energy performance of an open-joint ventilated façade compared with a conventional sealed cavity façade. Solar Energy 2011;85:1851–63.
[89] Baldinelli G. Double skin façades for warm climate regions: Analysis of a solution with an integrated movable shading system. Building and Environment 2009;44:1107–18.
[90] Manz H, Frank Th. Thermal simulation of buildings with double-skin façades. Energy and Buildings 2005;37:1114–21.
[91] Srebric J, Chen Q, Glicksman LG. A coupled airflow-and-energy simulation program for indoor thermal environmental studies. ASHRAE Transactions 2000;106:465–76.
[92] Zhai Z, Chen Q, Haves P, Klems JH. On approaches to couple energy simulation and computational fluid dynamics programs. Building and Environment 2002;37:857–64.
[93] Chen Q, van der Kooi J. ACCURACY—a computer program for combined problems of energy analysis, indoor airflow, and air quality. ASHRAE Transactions 1988;94:196–214.
[94] Zhai ZJ, Chen QY. Performance of coupled building energy and CFD simulations. Energy and Buildings 2005;37:333–44.